A COURSE ON
PART PROGRAMMING
PART PROGRAMMING

ACE DESIGNERS LTD.
NO. 533, X MAIN, IV PHASE,
PEENYA INDUSTRIAL AREA,
BANGALORE - 560 058.
# PART PROGRAMMING MANUAL

## CONTENTS

<table>
<thead>
<tr>
<th>I. INTRODUCTION</th>
<th>PAGE NO.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. INTRODUCTION</td>
<td>3</td>
</tr>
<tr>
<td>2. PROGRAM STRUCTURE</td>
<td>4</td>
</tr>
<tr>
<td>3. PROGRAM BLOCKS</td>
<td>7</td>
</tr>
<tr>
<td>4. PROGRAM FORMAT</td>
<td>7</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>II. AXES CO-ORDINATE SYSTEM, METHODS OF DIMENSIONING, MACHINE CO-ORDINATE SYSTEMS</th>
<th>PAGE NO.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. CO-ORDINATE AXES</td>
<td>11</td>
</tr>
<tr>
<td>2. DIRECTION OF MOVEMENT</td>
<td>12</td>
</tr>
<tr>
<td>3. METHODS OF DIMENSIONING</td>
<td>13</td>
</tr>
<tr>
<td>A. ABSOLUTE SYSTEM OF DIMENSIONING</td>
<td>14</td>
</tr>
<tr>
<td>B. INCREMENTAL SYSTEM OF DIMENSIONING</td>
<td>16</td>
</tr>
<tr>
<td>4. UNITS OF DIMENSIONING</td>
<td>17</td>
</tr>
<tr>
<td>5. MACHINE ORIGIN CO-ORDINATE SYSTEM</td>
<td>18</td>
</tr>
<tr>
<td>6. WORKPIECE CO-ORDINATE SYSTEM</td>
<td>18</td>
</tr>
<tr>
<td>7. MACHINE REFERENCE POINT CO-ORDINATE SYSTEM</td>
<td>18</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>III. PROGRAMMING OF G CODES, TOOL OFFSETS, CUTTER RADIUS COMPENSATION G CODE LIST</th>
<th>PAGE NO.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. G00 RAPID POSITIONING</td>
<td>19</td>
</tr>
<tr>
<td>2. G01 LINEAR INTERPOLATION</td>
<td>22</td>
</tr>
<tr>
<td>3. G02/G03 CIRCULAR INTERPOLATION * CIRCULAR INTERPOLATION PARAMETERS</td>
<td>23</td>
</tr>
<tr>
<td>4. G90 ABSOLUTE SYSTEM OF DIMENSIONING</td>
<td>25</td>
</tr>
<tr>
<td>5. G91 INCREMENTAL SYSTEM OF DIMENSIONING</td>
<td>31</td>
</tr>
<tr>
<td>6. G94 FEED RATE PER MINUTE</td>
<td>31</td>
</tr>
<tr>
<td>7. G95 FEED RATE PER REVOLUTION</td>
<td>31</td>
</tr>
<tr>
<td>8. G04 DWELL</td>
<td>38</td>
</tr>
<tr>
<td>9. G96 CONSTANT SURFACE SPEED</td>
<td>39</td>
</tr>
<tr>
<td>10. G92 MAXIMUM SPINDLE SPEED LIMIT</td>
<td>40</td>
</tr>
<tr>
<td>11. G97 CANCELLATION OF CONSTANT SURFACE SPEED</td>
<td>40</td>
</tr>
<tr>
<td>14. THREAD CUTTING</td>
<td>57</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>IV. M-CODE LIST</th>
<th>PAGE NO.</th>
</tr>
</thead>
<tbody>
<tr>
<td>59</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>V. MISCELLANEOUS FUNCTIONS</th>
<th>PAGE NO.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. CANNED CYCLES</td>
<td>61</td>
</tr>
<tr>
<td>2. DIRECT DIMENSION PROGRAMMING</td>
<td>75</td>
</tr>
</tbody>
</table>
Chapter – 1
Introduction
1. **INTRODUCTION**

1.1 **PROGRAM** :
To machine a component on a CNC lathe, informations like co-ordinate values and other technical data which indicate, how, a tool should be moved in relation to a work piece to achieve a desired machining form, is to be given in the form of coded instructions to the control unit. These informations are called a “PART PROGRAM”.

The data in the program block are namely :
a. Dimensional data (geometrical) and 
b. Control data
Control data contains informations like preparatory functions and auxiliary function data like turret index, coolant ON / OFF etc.,

1.2 **AXIS NOMENCLATURE** :
Standard right hand Cartesian co-ordinate system is used. Always cutting tool moves with respect to the work piece.
X-AXIS : Always parallel to the work-holding surface or perpendicular to main axis.
Z-AXIS : It is always the main spindle axis.
Refer figure 1 for the nomenclature.

C axis : Rotating axis around Z axis.
Text to be added.
Raw material Drawing

Machine drawing Change the drawing in absolute dimension. That is, input Machining co-ordinate

Machining plan Examines an efficient machining method.

Chuck form
- Soft or hard jaw
- Chucking width
- Manufacturing of jig if necessary
- Other special chucks

Tool selection
- Kind of cutting tools
- Number of cutting tools
- Kind of tips

Selection of Cutting conditions
- Determination of spindle speed
- Determination of feed
- Determination of cut depth
- Checking whether coolant oil is necessary

Set of G50 on each cutting tool Tooling diagram

Process sheet creating Write tool path and machine movement in accordance with the rule of NC machine.
NC program punching

---

Program check

---

Compare the indication of the current position indicator and each operation (M,S,T, function) of the machine with the process sheet to check the program.

1. Push “Program check button”
2. Operate the system by the mode of single block.
3. Cancel all cutter compensations.
4. Temporarily cut before determining the offset (compensation amount) of each tool.
5. Input the offset value corresponding to the offset number of each tool.
6. Determine the start point and idle the machine with the tool compensation applied to check interference with the chuck, etc.,

Machine operation

---

Measure the work and apply the compensation again if required.
2. **PROGRAM STRUCTURE:**
Part program comprises of a string of blocks written one after the other. When writing a program, a particular syntax is to be followed.

As per part program structure:
A. Program starts with program number.
B. Consists of series of blocks – dimensional and non-dimensional data like feed, speed, auxiliary functions etc.,
C. Program ends with program end code.

Character for program start, precedes the first block in the part program.
Character for main program start is given by “O”
To start program 1 – O 0001
To start program 2 – O 0002

Program start can also be called as program numbering. The program number is useful in identification of a program and calling a program.

**Similar to main program we have sub-routine program.** Sub-routine program is also called sub-program. A sub-program could contain a fixed sequence or frequently repeated pattern.

Character for a sub-routine program start is also given by “O”
To start sub-program 1 – O ........
To start sub-program 2 – O ........

Character for program end is written in the last block of the part program.
Character for main program end is M02 or M30.
Character for sub-program end is M99

**A sub-program can also call another sub-program upto two levels.**

Sub-program call:
Example: M98 P ........

No. of times Sub-program number.
Sub-program is called repeatedly

3. **PROGRAM BLOCKS:**
Program blocks contain data required to execute an operation. It is possible to write the program blocks with or without a sequence number. Sequence number is given at the block head. It need not be consecutive. It is used for the convenience of the operator. Sequence number is of 4 digits.

**Examples:**
* N0005 G00X20.Z5.; Program block written with block number.
  G00X20.Z5.; Program block written without block number.

All the blocks end with block end character ; or ★

If no block number is given or same block number is given to several blocks, then there cannot be a defined program jump or block search.
4. **WORD:**
   A program is made up a number of blocks. Similarly a block is made up a number of words.
   A word consists of an address character and a string of digits (alphanumeric character).
   An address character is normally an alphabet.
   X, Z, S, F ..... are all address characters.
   The string of digits may be specified with or without a sign and with a decimal point.
   The sign is written between the address letter and the string of digits. The positive sign may be omitted.

   **Examples of WORD:**
   1. X-243.127
      - String of digits
      - Sign (If it is positive it may be omitted)
      - Address character.
   2. Z 242.127
      - Here “+” sign is omitted
<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>N005</td>
<td>M07</td>
<td>Coolant on</td>
</tr>
<tr>
<td>N010</td>
<td>S500M03</td>
<td>Spindle on, clockwise direction of rotation, speed 500 rpm</td>
</tr>
<tr>
<td>N015</td>
<td>TXX00 ; (T0000)</td>
<td>Cancel tool offsets</td>
</tr>
<tr>
<td>N020</td>
<td>G00X…Z… ;</td>
<td>Rapid position to tool indexing point.</td>
</tr>
<tr>
<td>N025</td>
<td></td>
<td></td>
</tr>
<tr>
<td>N030</td>
<td>T0101 ;</td>
<td>Call tool 1 having tool offsets in 1 page</td>
</tr>
<tr>
<td>N035</td>
<td>F0.3 ;</td>
<td>Feed rate 0.3 mm / REV</td>
</tr>
<tr>
<td>N040</td>
<td>X26.Z2.</td>
<td>Position at φ 26 and 2mm in front of face (safe position to take cut)</td>
</tr>
<tr>
<td>N045</td>
<td>G01X40.Z5.</td>
<td>Linear interpolation</td>
</tr>
<tr>
<td>N050</td>
<td>Z-20. ;</td>
<td>Cut on OD.</td>
</tr>
<tr>
<td>N055</td>
<td>G02X50.Z-25.R5 ;</td>
<td>Circular interpolation, clockwise, radius 5mm</td>
</tr>
<tr>
<td>N060</td>
<td>G01Z-40.;</td>
<td>Linear interpolation, cut on ON</td>
</tr>
<tr>
<td>N065</td>
<td>X54. ;</td>
<td>Cut on face</td>
</tr>
<tr>
<td>N070</td>
<td>G00Z-38. ;</td>
<td>Rapid retraction</td>
</tr>
<tr>
<td>N075</td>
<td>M05 ;</td>
<td>Spindle speed zero rpm</td>
</tr>
<tr>
<td>N080</td>
<td>T0100. ; (T0000)</td>
<td>Cancel tool offsets</td>
</tr>
<tr>
<td>N085</td>
<td>G00X…Z… ;</td>
<td>Go to tool indexing point</td>
</tr>
<tr>
<td>N090</td>
<td>M05 ;</td>
<td>Spindle stop</td>
</tr>
<tr>
<td>N100</td>
<td>M09 ;</td>
<td>Coolant off</td>
</tr>
<tr>
<td>N105</td>
<td>M30 ;</td>
<td>End of program</td>
</tr>
</tbody>
</table>
5. **BLOCK FORMAT**:
The syntax followed for writing a program block should be as simple as possible. An example given below shows the program syntax that can be followed.

**Block example**:
N0015G00X20.Z40.F0.3S500T1M03 ;
N - Address of block number
G - Preparatory function
X,Z - Axes address
F - Feed rate
S - Spindle speed
T - Tool number
M - Miscellaneous function
; - Block end character

A. **BLOCK SKIP**:
Program blocks which need not be executed during every program run can be skipped by entering the slash character (/) in front of the sequence number with “BLOCK SKIP” switch in “ON” position.

**Example**:
N0055 G00X50Z2 ;
/N0060 G01Z-2 ;
/N0065 X40 ;
/N0070 G00Z2 ;
/N0075 X50 ;
Chapter-2
Axes Co-Ordinate System
Methods of Dimensioning
Machine Co-Ordinate System
Before going further into programming it is necessary to know some fundamental details regarding directions of tool movement, sign convention followed for directions of tool travel and axes co-ordinate system, different methods of dimensioning, units of dimensioning system etc.,

1. CO-ORDINATE AXES:
The directions of movement in a machine tool are based on a co-ordinate system allocated to the axes of motion of the slides.

As in the conventional lathe there is movement of slides in two directions along two axes, one axis being parallel to the spindle axis and the other axis perpendicular to the spindle axis.

The axis parallel to the spindle axis is called Z-axis. The axis perpendicular to the spindle axis is called X-axis.

Since the machine co-ordinate system has two axes, X and Z, it is referred to as the X-Z co-ordinate system.

MACHINE ORIGIN:
It is considered that the spindle nose front face is lying on X-axis and spindle axis as Z-axis thus the intersecting point of the two axes is the machine origin.
2. **DIRECTIONS OF MOVEMENT**:
Consider that a tool is positioned in I quadrant of X-Z co-ordinate system.

Conventionally speaking,

Along Z-axis any movement towards the spindle is considered as movement in the negative direction and any movement away from the spindle is considered as movement in the positive direction.

Along X-axis any movement towards the operator is considered as movement in the negative direction and any movement away from the operator is considered as movement in the positive direction.
3. METHODS OF DIMENSIONING:
There are two methods of dimensioning,

a. Absolute system of dimensioning or fixed zero system of dimensioning
b. Incremental system of dimensioning or floating zero system of dimensioning

ABSOLUTE SYSTEM OF DIMENSIONING OR FIXED ZERO SYSTEM OF DIMENSIONING:
In the figure, X and Z axes are divided into equal increments on either side of the origin.

To represent a point “P₁” which is located at 3 divisions to the right of the origin along Z-axis and 4 dimensioning above the origin along X-axis, it is written as P₁(X₄,Z₃). The positive value for X and Z is given because the point is located in the positive direction with respect to origin in both the axes.

Similarly to represent P₂, P₂ is written as P₂(X₅,Z₋₃) as it is in the positive direction along X-axis and in the negative direction along Z-axis. Thus all the points are referred to the fixed origin. The origin is also called as ABSOLUTE ZERO or FIXED ZERO. Since all the co-ordinate are defined with respect to fixed zero, this system is called as FIXED ZERO SYSTEM OF DIMENSIONING.
3.2 **INCREMENTAL SYSTEM OF DIMENSIONING OR FLOATING ZERO SYSTEM OF DIMENSIONING:**

With incremental system of dimensioning, target point co-ordinates do not refer to the fixed zero. Target point co-ordinates will be defined with respect to present position.

Assuming that tool is positioned at point $P_0$, if a reference is made for point $P_1$, co-ordinates of $P_1$ are defined with respect to point $P_0$.

Co-ordinates of $P_1$ with respect to $P_0$ are $X4Z3$.

Similarly if a reference is made for point $P_2$, co-ordinates of $P_2$ with respect to $P_1$ are $X4Z5$.

Again if a reference is made for $P_0$, co-ordinates of $P_0$ with respect to $P_2$ are $X-8Z-8$.

Thus incremental system doesn’t have a fixed origin. The origin changes from point to point. In other words the origin is floating and hence **FLOATING ZERO SYSTEM OF DIMENSIONING**.

![Diagram](image_url)

Co-ordinates of $P_1$ with respect to $P_0$ are $X4Z3$

Co-ordinates of $P_2$ with respect to $P_1$ are $X4Z5$

Co-ordinates of $P_0$ with respect to $P_2$ are $X8Z8
4. **UNITS OF DIMENSIONING**: 
Irrespective of whether absolute or incremental system is followed for dimensioning a component, it is possible to use both inch system and metric system of units.

With inch system all dimensions will be considered as in inches.

With metric system all dimensions will be considered as in mms.

**Metric system is followed in this manual.**

**NOTE:**
1. Generally all the programs are written in absolute system.
2. All the programmed X values will be in diameters.
Different co-ordinate systems and reference points used on a CNC machine are explained below.

5. **MACHINE ORIGIN CO-ORDINATE SYSTEM**:
   Machine origin co-ordinate system is the axes co-ordinate system having the origin at machine zero.

   **MACHINE ZERO – M**:
   The points of intersection of the spindle axis and the plane of spindle front face is taken as machine zero.

   Similar to machine co-ordinate system, we have work piece co-ordinate system having the origin at work zero and the reference point co-ordinate system having the origin at machine reference point.
6. WORK PIECE CO-ORDINATE SYSTEM:
This is the co-ordinate system having the origin at work piece zero. This co-ordinate system is used for the purpose of programming co-ordinates of different points on the work piece.

WORK PIECE ZERO – W:
Work piece zero is the zero defined for programming the work piece dimensions. It can be freely selected.

The coinciding point of the work piece front face and the work axis is selected as work zero.

7. MACHINE REFERENCE POINT CO-ORDINATE SYSTEM:
This is the machine co-ordinate system having the origin at machine reference point.

MACHINE REFERENCE POINT – R:
There is a difference between machine origin and machine reference point. A programmer has to know this part clearly.

The reference point R is a point defined by the machine manufacturer which is to be approached when the control is switched on and which synchronizes the system. The reference dimension is defined in the machine data.

With the machine co-ordinate system having the origin at reference point, whenever the slides are taken to reference point the display values of the actual position (on the CRT screen) will be zero.

With machine co-ordinate system having the origin at machine zero, reference point co-ordinates are to be defined with respect to machine zero. For all the actual requirements keeping the slides at the machine reference point, distance between the machine zero and the turret zero (turret front face and axes intersecting point) is established in both the axes and defined as the reference point co-ordinates.

PARKING POSITION – T:
This is an intermediate or an arbitrary point on the machine which is defined by the operator. The position is selected such that, it would be convenient for tool indexing, inspection of component or tool, loading and unloading of component etc.,
Chapter-3
Programming of G-CODES,
Zero offsets, Tool offsets &
(Tool Nose) Radius Compensation
PROGRAMMING OF G CODES:
Command data given to the CNC system in the form of part program comprises of a set of numerical codes.

G codes are one among those codes.

G codes are also called as preparatory functions.

They take active part in part program execution and machine operation and are programmed by letter G followed by two digits as per format G2. They are always programmed at the start of the block. They describe the type of machine movement, type of interpolation, type of dimensioning, time related functions and activate certain operating conditions within the control.

The action of G codes is either modal or block by block.

G codes once programmed, remain active until another G code of the same group is programmed, after which the previous one gets cancelled, are said to be MODAL.

G code which remains active only in the block in which it is programmed, is said to be BLOCKWISE ACTIVE or ONE SHOT G CODE.

G codes which are active after switching on controller or reset or M02 or M30 are called SYSTEM RESET’s or DEFAULT “G” codes. They need not be programmed unless they are replaced by a G code of the same group other than resets.
# PART PROGRAMMING MANUAL

## G-CODE LIST:

<table>
<thead>
<tr>
<th>G-CODE</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00</td>
<td>Rapid positioning – (rapid traverse)</td>
</tr>
<tr>
<td>G01</td>
<td>Linear interpolation – (straight line cutting)</td>
</tr>
<tr>
<td>G02</td>
<td>Circular interpolation – clockwise</td>
</tr>
<tr>
<td>G03</td>
<td>Circular interpolation – counter clockwise</td>
</tr>
<tr>
<td>G04</td>
<td>Dwell in seconds</td>
</tr>
<tr>
<td>G20</td>
<td>Inch input (0.0001 in)</td>
</tr>
<tr>
<td>G21</td>
<td>Metric input (0.001 mm)</td>
</tr>
<tr>
<td>G33</td>
<td>Thread cutting</td>
</tr>
<tr>
<td>G40</td>
<td>Tool nose radius compensation – cancel</td>
</tr>
<tr>
<td>G41</td>
<td>Tool nose radius compensation – left</td>
</tr>
<tr>
<td>G42</td>
<td>Tool nose radius compensation – right</td>
</tr>
<tr>
<td>G90</td>
<td>Absolute programming</td>
</tr>
<tr>
<td>G91</td>
<td>Incremental programming</td>
</tr>
<tr>
<td>G92</td>
<td>Spindle speed limitation with CSS</td>
</tr>
<tr>
<td>G94</td>
<td>Feed per minute</td>
</tr>
<tr>
<td>G95</td>
<td>Feed per revolution</td>
</tr>
<tr>
<td>G96</td>
<td>Constant surface speed (CSS)</td>
</tr>
<tr>
<td>G97</td>
<td>CSS cancel</td>
</tr>
</tbody>
</table>

## CANNED CYCLES:

| G70    | Finishing cycle |
| G71    | Stock removal in turning |
| G72    | Stock removal in facing |
| G73    | Pattern repeating |
| G74    | Peck drilling |
| G76    | Multiple threading cycle |
**G00 RAPID POSITIONING:**
Position data command (target point co-ordinates) programmed after the code G00 is traversed at the maximum possible feed rate along a straight line (at rapid rate) set by the machine tool manufacturer for each axis independently (parameter 0518 ～ 0521).

G00 can be programmed as G or G0. G00 is modal. It gets cancelled with G01, G02, G03 or G33.

With G00 being effective, the programmed feed rate gets cancelled, until G01, G02 or G03 is programmed.

Assume that the tool is positioned at X20, Z0.

To move from 1 to 2, command data given is G00 X40.Z-20 ;
From 2 to 3  G00Z-40 ; is programmed

On the CRT screen, movements executed with G00 appear as dotted lines if graphics feature is available.

To the possible extent all idle movements are programmed with G00.

**For example:**
1. Movement from machine reference point to turret index position.
2. Movement from turret index point to safe positioning point before cutting and after cutting.
3. All idle movements when the tool is away from component.
G01 LINEAR INTERPOLATION:
Position data command programmed after G01 is executed at the programmed (or at the set) feed rate, the movement being made along a straight line joining start point and final point. Control calculates the tool path by linear interpolation.

Whether one axis is moved at a time or simultaneously two axes are moved, the movement will be along a straight line.

CNC calculates the feed rate of each axis so that the resultant feed rate is the programmed feed rate.

The feed rate specified by “F” code remains effective until a new feed rate is commanded. Hence it need not be repeated. If feed rate if not specified it will be regarded as zero. The movement command can be in absolute or incremental command.

Assume that tool is positioned at X40 Z0. It has to reach point 2 and then point 3.

Feed rate being 0.2 mm/rev., command data given is:
From 1 to 2  G01X60.Z-20.F0.2 ; negative movement in both axes
From 2 to 3  G01Z-40. ; negative movement in one axes.
When the tool is moving from point 2 to point 3 there is no need to define the X value as the movement is along Z-axis.

G01  X60.Z-20.  F0.2  

End of block
Programmed feed rate 0.2 mm/rev
Position data
Movement at set feed rate

G01 is the reset mode. It is modal and gets cancelled with G00, G02, G03 or G33.

On the graphic screen movements made with G01 appear as continuous lines.
**G02/G03 CIRCULAR INTERPOLATION:**
Position command data given after G02 or G03 is executed along a circular path at the set feed rate.

Circular interpolation effects a tool movements along a circular arc:
- In a clock wise direction with G02
- In a counter clockwise direction with G03

When a position command is to be given, it is necessary to know.
1. The start point of the circular arc
2. The end point of the circular arc
3. The radius of the arc or
4. The circular interpolation parameters (explained in the next section)

Start point of the circular arc will be known as it is the end point of the preceding command in the program.

Radius will be given in the component drawing.

Feed rate for circular interpolation is specified by “F”
End point is to be calculated depending on the component profile. Format to be used to give a circular interpolation command with radius is as below.

- **G02/G03**
- **X…Z**
- **R**
- **F**

  - **Feedrate**
  - **Arc Radius**
  - **End point co-ordinates**
  - **Direction of interpolation**

**Example:**
- G02X50.Z-25.R5 F0.03  - C.W.
- G03X-100.Z-37.R8 F0.03  - C.C.W
CIRCULAR INTERPOLATION PARAMETERS OR CIRCLE POINT CO-ORDINATES:
The centre point is defined by either the radius or by the circular interpolation parameter I and K.

The interpolation parameters are the paraxial co-ordinates of the distance vector from the starting position to the centre point of the circle. The interpolation parameters I and K are allocated to X and Z axes respectively.

From point 1 to point 2 the tool travels at an angle. From 2 to 3 it travels in a circular path. It is possible to establish the offsets of the centre point with respect to the start point in both the axes. The x-axis offset is I and the Z axis offset is K.

To move from point 2 to point 3, the following program is given:
N0015 G02X3.Z3.I…K…F….;

Instead of radius R, the interpolation parameters I and K could be given to define the centre point. The interpolation parameters must always be entered as incremental values irrespective of absolute or incremental programming is done. The sign is based on the direction of co-ordinate from the starting position to the centre point of the circle.
See the following examples where,
S = Start point
E = End point
C = Centre point

As explained above the programming format for the four arcs can be written as

1. G02 X…Z…I… K-…F
2. G02 X…Z…I-…K-…F
3. G02 X…Z…I….K….F
4. G03 X…Z…I… K-…F

G02 and G03 are modal and get cancelled with each other and also with G00, G01 and G33. The following table in a nutshell explains about the circular interpolation parameters.
### PART PROGRAMMING MANUAL

<table>
<thead>
<tr>
<th>SL. NO.</th>
<th>DATA TO BE GIVEN</th>
<th>COMMAND</th>
<th>MEANING</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Rotation direction</td>
<td>G02</td>
<td>Clockwise direction (CW)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>G03</td>
<td>Counter clockwise direction (CCW)</td>
</tr>
<tr>
<td>2</td>
<td>End point position</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Absolute command</td>
<td>X,Z</td>
<td>End point position in the work coordinate system.</td>
</tr>
<tr>
<td></td>
<td>Incremental Command</td>
<td>U,W</td>
<td>Distance from start point to end point.</td>
</tr>
<tr>
<td>3</td>
<td>Distance from start point to</td>
<td>I,K</td>
<td>Distance with direction from start point to arc center. (Always radius value and incremental value).</td>
</tr>
<tr>
<td></td>
<td>center</td>
<td>R</td>
<td>Radius of arc. (Always a radius value).</td>
</tr>
<tr>
<td>4</td>
<td>Feedrate</td>
<td>F</td>
<td>Feedrate along the arc.</td>
</tr>
</tbody>
</table>

4. **INCH / METRIC CONVERSION:**
   
   Either inch or metric input can be selected by a G code.

<table>
<thead>
<tr>
<th>UNIT SYSTEMS</th>
<th>G-CODE</th>
<th>LEAST INPUT INCREMENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inch</td>
<td>G20</td>
<td>0.0001 inch</td>
</tr>
<tr>
<td>Metric</td>
<td>G21</td>
<td>0.001 mm</td>
</tr>
</tbody>
</table>

- This G code must be specified in an independent block before setting the co-ordinate system at the beginning of the program.
- When the power is turned ON the NC status is the same as that held before the power was turned OFF.
- G20 / G21 must not be switched during program.
- When switching inch to metric and vice versa, the offset value must be reset according to the input unit.
When the diameter is specified, it is called diameter programming and when the radius is specified, it is called radius programming. Radius programming or diameter programming can be specified by parameter (no. 0019, XRC). When using diameter programming on the X-axis, note the conditions listed in the following table.

<table>
<thead>
<tr>
<th>ITEM</th>
<th>NOTES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z axis command</td>
<td>Specified irrespective of diameter or radius value.</td>
</tr>
<tr>
<td>X axis command</td>
<td>Specified with a diameter value.</td>
</tr>
<tr>
<td>Incremental command with address U</td>
<td>Specified with a diameter value. In the above figure, specifies form D₂ to D₁ for tool path B to A.</td>
</tr>
<tr>
<td>Co-ordinate system setting (G50)</td>
<td>Specifies a X-axis co-ordinate value with a diameter.</td>
</tr>
<tr>
<td>X component of tool offset value</td>
<td>Parameter setting (no. 0001, ORC) determines either diameter or radius value.</td>
</tr>
<tr>
<td>Parameters in G90-G94, such as cutting depth along X-axis (R )</td>
<td>Specifies a radius value.</td>
</tr>
<tr>
<td>Radius designation in circular interpolation (R,I,K)</td>
<td>Specifies a radius value.</td>
</tr>
<tr>
<td>Feedrate along x-axis</td>
<td>Change of radius / rev.</td>
</tr>
<tr>
<td></td>
<td>Change of radius / min.</td>
</tr>
<tr>
<td>Display of x-axis position</td>
<td>Display as diameter value.</td>
</tr>
</tbody>
</table>
5. **ABSOLUTE / INCREMENTAL PROGRAMMING:**
   These are two ways of command for axis movement. In absolute command, co-ordinate value of the end point is programmed. In incremental command distance to be moved is programmed. G90 is used for absolute command and G91 is used for incremental command.
   - Absolute and incremental commands can be used together in a block
     eg:    X400. W-400. 
           U 200. W-400.0 – Incremental command.

6. **G94 FEED RATE PER MINUTE:**
   When the code G94 is programmed CNC assumes that feed rate value entered under address F is in mm / min (or inch / min)
   G94F100. ; 100 mm / min
   G94G20F10. ; 10 inches / min.
   G94 is modal. It is cancelled with G95.

7. **G95 FEED RATE PER REVOLUTION:**
   When the code G95 is programmed CNC assumes the feed rate value entered under address F is in mm/ rev. (or inch / rev.)
   G95F0.2 ; 0.2 mm/rev
   G95G20F0.01 ;

   G95 is default and modal. It is cancelled with G94.

   To have G95 effective spindle speed encoder is provided. Spindle encoder sends the feedback signal of spindle speed to the system, which in turn, controls the feed rate to have a constant feed per revolution.
# PART PROGRAMMING MANUAL

<table>
<thead>
<tr>
<th>Meaning</th>
<th>FEED PER MINUTE</th>
<th>FEED PER REVOLUTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Amount of movement per minute</td>
<td>Amount of movement per revolution of spindle</td>
<td></td>
</tr>
<tr>
<td>Designated address</td>
<td>F</td>
<td>F</td>
</tr>
<tr>
<td>Designated G code</td>
<td>G94</td>
<td>G95</td>
</tr>
<tr>
<td>Range of designation</td>
<td>Metric input</td>
<td>Over ride</td>
</tr>
<tr>
<td>1 mm/min to 100000 mm/min (F1 to F100000)</td>
<td>0.001 mm/rev to 500.0000 mm/rev (F1 to F5000000)</td>
<td></td>
</tr>
<tr>
<td>Increment system 1/10</td>
<td>1 mm/min to 02000 mm/min (F1 to F02000)</td>
<td>0 to 150% over ride can be applied (in increments of 10%)</td>
</tr>
<tr>
<td>Inch input</td>
<td>0.01 to 4000.00 inch/min (F1 to F4000000)</td>
<td></td>
</tr>
<tr>
<td>Increment system 1/10</td>
<td>0.01 inch/min to 480.00 inch/min (F1 to F47000)</td>
<td></td>
</tr>
<tr>
<td>Clamping value</td>
<td>Clamping takes place at a certain specific speed for both feed per minute and feed per revolution.</td>
<td></td>
</tr>
<tr>
<td>Over ride</td>
<td>For both feed per minute and feed per revolution.</td>
<td></td>
</tr>
</tbody>
</table>
EXAMPLE 1:

<table>
<thead>
<tr>
<th>POSITION</th>
<th>X</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>20</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>20</td>
<td>-10</td>
</tr>
<tr>
<td>3</td>
<td>40</td>
<td>-10</td>
</tr>
<tr>
<td>4</td>
<td>40</td>
<td>-20</td>
</tr>
<tr>
<td>5</td>
<td>60</td>
<td>-20</td>
</tr>
<tr>
<td>6</td>
<td>60</td>
<td>-30</td>
</tr>
</tbody>
</table>

Considering a movement from point 1 to 6 via points 2,3,4 and 5, program could be written as:

N5 G0 X20..Z2 ; (Safe positioning to take cut)
N10 G01Z-10. F0.25 ; (Movement to point 2)
N15 X40. ; (From point 2 to point 3)
N20 Z-20 ; (From point 3 to point 4)
N25 X60. ; (From point 4 to point 5)
N30 Z-30 ;

In block number N10, G00 is replaced by G01. Since G01 is modal it is effective throughout the program.
EXAMPLE 2 :

<table>
<thead>
<tr>
<th>POSITION</th>
<th>X</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>24</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>30</td>
<td>-3</td>
</tr>
<tr>
<td>3</td>
<td>30</td>
<td>-22</td>
</tr>
<tr>
<td>4</td>
<td>36</td>
<td>-25</td>
</tr>
<tr>
<td>5</td>
<td>50</td>
<td>-25</td>
</tr>
<tr>
<td>6</td>
<td>61.592</td>
<td>-29.447</td>
</tr>
<tr>
<td>7</td>
<td>70</td>
<td>-45</td>
</tr>
<tr>
<td>8</td>
<td>70</td>
<td>-55</td>
</tr>
<tr>
<td>9</td>
<td>80</td>
<td>-60</td>
</tr>
<tr>
<td>10</td>
<td>94</td>
<td>-60</td>
</tr>
<tr>
<td>11</td>
<td>100</td>
<td>-63</td>
</tr>
<tr>
<td>12</td>
<td>100</td>
<td>-75</td>
</tr>
</tbody>
</table>

Use of G02/G03, G01 and G00 is given clearly. They are all modal and replace each other.
EXAMPLE 3:

<table>
<thead>
<tr>
<th>POSITION</th>
<th>X</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>20</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>20</td>
<td>-30</td>
</tr>
<tr>
<td>3</td>
<td>43.094</td>
<td>-50</td>
</tr>
<tr>
<td>4</td>
<td>51.754</td>
<td>-52.5</td>
</tr>
<tr>
<td>5</td>
<td>70</td>
<td>-52.5</td>
</tr>
<tr>
<td>6</td>
<td>70</td>
<td>-100</td>
</tr>
<tr>
<td>7</td>
<td>51.754</td>
<td>-47.5</td>
</tr>
</tbody>
</table>

\[ I = +4.33 \quad K = +2.5 \]

N5 G00 X20.Z2 ;
N10 G01 Z-30..F0.2 ;
N15 X43.094 Z-50. ;
N20 G02 X51.754 Z-52.5 I4.33 K2.5 ;
N25 G01 X70. ;
N30 Z-100. ;

In this example the circular interpolation is programmed using interpolation parameters instead of the radius.

The I & K values are given as incremental data. Since the centre point is located in positive direction in both the axes with respect to the start point they have positive values.
PROCEDURE FOR TAKING OFFSET:

1. Make sure that both the axes are referenced and the component is securely clamped.
2. Now bring one of the tools near the face of the job.
3. Select MDI mode.
4. Press PROGRM Button
5. Enter S500, EOB and then press Insert.
6. Press CY START PB.
7. Select Handle / Jog mode and select the appropriate feed.
8. Press spindle CW / CCW depending on the type of the tool.
9. Light facing cut to be taken up to the center.
10. After the finish cut, move the tool back in X only. DON’T DISTURB Z-AXIS.
11. Now switch off the spindle.
12. Press OFFSET SETTING Soft key on the CRT. The OFFSET / SETTING / WORK are displayed on the CRT.
13. Press OFFSET Soft Key, WEAR / GEOM are displayed on the CRT.
14. Press GEOM Soft Key and position the cursor using cursor movement buttons on the required offset number corresponding to the tool used.
15. Press Z 0 and press MEASUR Soft Key. The Z-offset for the selected tool is set.
16. Now rotate the spindle in the appropriate direction and machine on OD.
17. DO NOT MOVE X.
18. Take Z-axis away from the job.
19. Stop the spindle.
20. Measure the OD dimension.
21. Press GEOM Soft Key and position the cursor using cursor movement buttons on the required offset number corresponding to the tool used.
22. Press X, the measured value and then press MEASUR Soft Key. The X-Offset for the tool is set.
23. Repeat the procedure for all tools.
24. After taking offsets, select MDI and command S0
EXAMPLE 4:

<table>
<thead>
<tr>
<th>POSITION</th>
<th>X</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>10</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>20</td>
<td>-5</td>
</tr>
<tr>
<td>3</td>
<td>20</td>
<td>-10</td>
</tr>
<tr>
<td>4</td>
<td>30</td>
<td>-15</td>
</tr>
<tr>
<td>5</td>
<td>30</td>
<td>-25</td>
</tr>
<tr>
<td>6</td>
<td>50</td>
<td>-25</td>
</tr>
<tr>
<td>7</td>
<td>50</td>
<td>-30</td>
</tr>
</tbody>
</table>

N10 G00 X10.Z5.;
N15 G94 G01 Z0F50.;
N25 Z-10.;
N30 G95 G02 X30.Z-15.R5. F0.21;
N35 G01 Z-25.;
N40 X55.
N45 G00 Z5.;

The above example gives a changeover from feed / rev. to feed / min.

N15 G94 G01 Z0.F50.;

The code G94 activates feed / min. mode and F50 refers to 50 mm/min.

N30 G95 G02 X30.Z-45.R5 F0.2;

The code G95 activates feed / rev. mode and F0.2 indicates 0.20 mm/rev.
9. REFERENCE POINT:
The reference point is a fixed position on a machine, to which the tool can be easily moved by the reference point return or zero return function.

* Automatic reference point return (G28):
This command is used to position the axes at reference point at rapid traverse. Generally used for tool change / turret index. The reference point lamp goes “ON”. For safety the cutter compensation and tool length compensation should be cancelled before executing this command.

* Reference point return check (G27):
This function checks whether the program returns the tool to the reference point. The tool is positioned at rapid traverse rate. If the tool reaches the reference point, the reference point return lamp turns ON. If the position reached is not the reference point an alarm is displayed.

1. G04 DWELL:
G04 is used to program dwell time in seconds. The dwell time value is programmed by means of letter X or P or U or Z or F.

The time duration for dwell is as follows.

To have a dwell of 2 seconds.
N20 G04 X2. ; is programmed.

Also, it can be programmed as
N20 G04 U2.1 ; OR N20 G04 P2000 ;

No other function can be programmed in a block in which G04 is programmed. G04 is blockwise active. The maximum command time is 99999.999 seconds. Error for the item is within 16 msec.

Dwell starts after the commanded feedrate of the previous block attains zero.
2. G96 CONSTANT SURFACE SPEED (CSS):
In this case surface speed is set by S. The spindle speed is calculated so that surface speed is maintained as specified in correspondence with tool position. Even when the work piece diameter is changed, the CNC changes the spindle speed so that cutting speed remains constant. This is called constant surface speed function.

Code G96 indicates the constant surface speed in m/min. i.e. S value programmed after G96 is considered as meters / min.

EXAMPLE:
N5 G96 S500 ; 500 m/min
N10 G96 S200 ; 200 m/min

The CNC calculates the spindle speed for the current turning diameter in accordance with the programmed cutting speed.

The speed value stored under S, remains unchanged until a new speed value is programmed. i.e. S value programmed is modal.

When CSS is used the work co-ordinate system must be set so that the centre of rotation meet the Z-axis (X=0)

G96 is modal. It cancels with G97.

- The surface speed for a block containing rapid traverse is calculated on the basis of the end point.
- When the power supply is turned ON, the maximum spindle speed is not set; hence the speed is not clamped.
- G50S0 indicates speed is clamped at 0 rpm.
- The “S” value specified in G96 mode is unaffected by G97 and hence is restored when returned to G96 mode.

The surface speed specified in G96 is for the program path and not for the position with tool.
12. **G92 SET VALUE OF MAXIMUM SPINDLE SPEED LIMIT:**

   G92 is the set value of maximum spindle speed limit in rpm, when working with constant surface speed.

   - \texttt{G92S500} ; Maximum spindle speed of 500 rpm.
   - \texttt{G92 S2000} ; Maximum spindle speed of 2000 rpm.

   G92 is given to safe guard the spindle from running at high rpm values when working at smaller diameters under constant surface speed. Spindle speed set by G92 can not be overrun when operating in constant surface speed mode or by using spindle speed override keys.

   The spindle speed limit value stored under address S with G92 remains in the program until a new spindle speed limit value is programmed. \texttt{G92SO}, stops the spindle.

   Spindle speed values programmed with G92 are modal. No other information can be programmed in this block.

13. **G97 CANCELLATION OF CONSTANT SURFACE SPEED:**

   G97 is the cancellation of constant cutting speed and storing of final speed set point of G96 i.e. after programming G97, the speed at which the spindle was running (when working under constant surface speed) is retained as the constant spindle speed.

   \textbf{EXAMPLE}:
   \begin{verbatim}
   N5 G96 S250 ;
   N10 G01 X26.Z5.F0.21 ;
   N15 Z-25. ;
   N20 X30. ;
   N25 G00 Z5. ;
   N30 G97 ;
   \end{verbatim}

   N5 – constant cutting speed 250 m/min.

   N30 – G97 cancels G96. Spindle speed correspondence to X30 (2650 rpm) is retained.

   No other information is programmed in this block.
EXAMPLE 6:

<table>
<thead>
<tr>
<th>POSITION</th>
<th>X</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>40</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>40</td>
<td>-15</td>
</tr>
<tr>
<td>3</td>
<td>40</td>
<td>-25</td>
</tr>
<tr>
<td>4</td>
<td>40</td>
<td>-35</td>
</tr>
<tr>
<td>5</td>
<td>60</td>
<td>-35</td>
</tr>
<tr>
<td>6</td>
<td>60</td>
<td>-50</td>
</tr>
</tbody>
</table>

At the bottom of grooves a dwell of one second is to be programmed.

N20 G00 X45.Z-15. ;
N25 G01 X30.F0.2 ;
N30 G04 X1. ; (Dwell of 1 second)
N35 G00 X45. ;
N40 Z-25. ;
N45 G01 X30. ;
N50 G04 X1. ; (Dwell of 1 second)
N55 G00 X45. ;

Block numbers N25 and N45 correspond to grooving operation. At the bottom of each groove a dwell of one second is programmed. Block numbers N30 and N50 represent the same.
EXAMPLE:

<table>
<thead>
<tr>
<th>POSITION</th>
<th>X</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>25</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>25</td>
<td>-50</td>
</tr>
<tr>
<td>3</td>
<td>55</td>
<td>-50</td>
</tr>
<tr>
<td>4</td>
<td>55</td>
<td>-75</td>
</tr>
<tr>
<td>5</td>
<td>90</td>
<td>-75</td>
</tr>
<tr>
<td>6</td>
<td>90</td>
<td>-90</td>
</tr>
</tbody>
</table>

Assume a finishing stock of 0.1mm on all faces, and 0.2mm on all diameters. A roughing cut of 1.4mm is to be given on $\phi$25. Roughing feed rate is 0.3 mm/rev and finishing feed is 0.15 mm/rev.

Roughing is done at 1500 rpm. Finishing is done at a constant surface speed of 125 mts/min with the maximum spindle speed limited to 1800 rpm.

In the above program, block number N45 corresponds to constant surface speed of 125 mts/min and block number N50 corresponds to limitation of maximum rpm 1800 rpm. Block number N95 – change over from constant surface speed to constant rpm mode.
19. **ZERO OFFSETS:**
G codes right from G54 to G59 are called zero offsets. Zero offsets are used to define a new origin with respect to machine zero.

There are two types of zero offsets:

a. Settable zero offsets or adjustable zero offsets. G54, G55, G56, G57 are the settable zero offsets.

b. Programmable zero offsets. G58 and G59 are the programmable zero offsets.

19.1 **SETTABLE ZERO OFFSETS:**
Zero offset data of the new origin can be directly entered into these zero offset memory locations. The stored offset values can be altered any time.

The offset values of the new origin can be entered into any of the zero offset locations. When the relevant zero offset number is selected in the program, CNC refers to the corresponding zero offset location and shifts the machine zero to the new origin position by the amount of offset value stored for each axis.

**EXAMPLE:**
N50 G55 ;

When CNC reads the block number N50 the machine origin gets shifted by the amount of offset value stored in G55 location.

When zero offset data is selected, by using the soft key menu, CRT displays the following data.

X Z (G54)
X Z (G55)
X Z (G56)
X Z (G58)

Offset data of the new origin is stored into any of the above zero offset memory locations.

Generally settable zero offsets are used to define the work zero. Since work piece axis is in line with spindle axis, there is no offset along X-axis, and since the work piece zero is in front of machine zero, it has a particular amount of offset along Z-axis.
A new origin \( O \) is defined in the figure. With respect to machine zero, the offsets of the new origin are \( X_0 \) \( Z_{200} \).

If the data (i.e. \( Z_0 \) \( Z_{200} \)) is stored in G57 memory location, whenever G57 is used in the program, offset data entered in G57 is considered by CNC.

Offset data stored in G57 is displayed as below,
\( X_0 \) \( Z_{200} \) (G57)

The tool is to be positioned at point \( P \) (\( X_{40} \), \( Z_{-30} \)) with respect to G57.

The program is,

\[ \text{N50 G57 ;} \]
\[ \text{N60 G00 X40 Z-30 ;} \]

To position tool at \( P \) from an existing position \( P_1 \), control calculates the distance to be traveled in both the axes considering the offset values of the present position with respect to machine zero, offset value of G57 origin with respect to machine zero, and offset position of \( P \) with respect to G57 origin.

Distance to be traveled from \( P_1 \) to position tool at \( P \):

Along X-axis =
\[ \text{X offset of } P \text{ with } + \text{ X offset of G57 origin with } - \text{ X offset of } P_1 \text{ with respect to G57 respect to machine zero respect to machine zero.} \]
\[ = 20 + 0 - 50 = -30 \]

Along Z-axis =
\[ \text{Z offset of } P \text{ with } + \text{ Z offset of G58 origin with } - \text{ Z offset of } P_1 \text{ with respect to G57 respect to machine zero respect to machine zero.} \]
\[ = 30 + 200 - 250 = -80 \]
19.2 PROGRAMMABLE ZERO OFFSETS:
These offsets are similar to settable offset values except that zero offset data cannot be
directly entered into these zero offset memory locations. In order to enter zero offset data
into these offset locations, the zero offset data is programmed under the relevant G code.
When the program is executed, after reading the block in which the zero data is
programmed, CNC transfers the offset values into the corresponding zero offset memory
location, and the machine zero gets shifted to the new origin position.

To enter offset data into programmable zero offsets the following program is written.
N50 G58 X0 Z200 ;

When CNC reads N50, the data programmed under G58 is loaded into G58 memory
location, and the machine zero gets shifted to new origin.

To cancel programmable zero offsets the following program is written.
N100 G58 X0 Z0 ;

Zero offset values are additive i.e. when more than one zero offset code (G code) are used
in a program, CNC takes sum of all the offset values into consideration, in order to shift
the machine zero to new zero position.

NOTE:
a. G54 is the default function i.e. there is no need to program G54 code, in order to
shift the machine zero to new position, when a zero offset data is entered into G54
memory location.
b. Also when a zero offset location other than G54 is used, in order to enter the zero
offset data, care should be taken to keep the G54 memory location empty. Otherwise, as G54 is the default, and the zero offsets are additive, whenever a G
code other than G54 is programmed, zero offset value stored in G54 memory
location is added to that value stored in the zero offset location of the programmed
G code.

20. G53 SUPPRESSION OF ZERO OFFSETS:
Zero offsets are modal. Command positions programmed after programming zero offsets,
are referred to the new origin. In case it is necessary to refer a position command with
respect to machine zero, suppressing the programmed zero offsets, then the code G53 is
programmed. G53 is block wise active. i.e. it can suppress zero offsets only in the block in
which it is programmed.
EXAMPLE:
N5 G55;
N10 G00 X100 Z200;
|   |
|   |
N50 G53 X100 Z200;
N55 X100 Z200;

Block number N5 - Activates the zero offset already loaded in the zero offset memory G55.
Block number N10 - Command position is with respect to new origin.
Block number N50 - G53 suppresses zero offset G55, and command position is with respect to machine zero.
Block number N55 - Command position is once again with respect to G55 zero.

In case of touch probe, G53 is used for automatic tool offset entry in JOG MODE, and automatic updating of already entered tool offsets in AUTOMATIC MODE (by executing a special program, where all command positions are to be defined with respect to machine zero).

21. TOOL PROGRAMMING – T:
Tool programming is done, to call the tools mounted on the turret in different stations. The tool programming is done by means of the code T.

21.1 TOOL NUMBER Tn.:
To the right of T any number from 1 to 8 is programmed. The number programmed to the right of T, indicates the turret station in which the tool is mounted (in case of 12 station turret, the number entered is from 1 to 12).

To call tool 1 (station 1)
N5 T01; is programmed.

21.2 CANCELLATION OF TOOL NUMBER:
Tool number can be cancelled by programming T00.

N5 T00; cancels tool number selected earlier.
22. **TOOL OFFSET OR TOOL LENGTH OFFSET:**

Tool offset is to compensate for the difference between the actual tool used and the imagined part. Tool offset or tool length offset is the offset of the turret zero with respect to tool tip.

Turret zero is the intersection point of turret axis and turret front face.

A tool is mounted on turret. The distance between the turret zero and the tool tip along X-axis is $X_T$, and that along Z-axis, is $Z_T$.

22.1 **TOOL OFFSET MEMORY LOCATION:**

Tool length offsets are entered into tool offset memory locations. There are 16/32 tool offset memory locations. Tool offsets are identified by the code XX AFTER TOOL NUMBER. Tool offsets are numbered from 1 to 32. Tool offsets for each tool are entered into separate tool offset memory locations. Established tool length data of a particular tool can be entered into any of the 32 tool offset memory locations.

To bring in the established tool offset values of a called tool, the corresponding offset number is to be programmed following the tool number.

Consider that for tool 1, the offset values are entered in tool offset memory location 1; Then, N50 T01 01; is programmed.
<table>
<thead>
<tr>
<th>TOOL OFFSET NO.</th>
<th>OFX</th>
<th>OFZ</th>
<th>RADIUS</th>
<th>TYPE</th>
</tr>
</thead>
<tbody>
<tr>
<td>01</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>32</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Tool number indicates the turret station number where the tool is mounted, and the tool lengths of which are to be entered.

Tool type indicates tool location code or tool identification code.

Of X indicates X offset length of tool.

Of Z indicates Z offset length of tool.

Dia / radius indicates tip radius of tool.

Wear 1 indicates tool wear along X-axis.

Wear 2 indicates tool wear along Z-axis.

Radial wear indicates radial wear.

**EXAMPLE:**
Tool number 5

X offset 180

Z offset 7

Data is to be entered into tool offset NO10.

Following data is entered into no. 10.
X TOOL OFFSET XT = X VALUE AT W/P – W/P RADIUS (ON RADIUS)

= 50-25

= 25
Z TOOL OFFSET ZT = Z VALUE WITH TABLE TOUCHING THE COMP. – Z VALUE WITH TOOL TOUCHING THE COMPONENT

= 200 – 175
= 25
Cursor moves down, line by line when “ENTER” key is pressed.

When ever the tool is called to a particular position, CNC takes into account, zero offset, tool length offset, distance of the target position from work zero, and the position where the tool is located and then calculates the distance to be traveled in each axis.

The position of tool tip with respect to turret zero is different for each tool. Tool offsets are used to establish, the offsets of each tool tip with respect to machine zero.

Assume that the machine reference point is X600:ZX500?

The tool is brought close to the component, and a cut is taken on OD. The diameter is measured. The corresponding display position in the X-axis is noted down. Assume that diameter measured is 50mm and the corresponding display position is X400.

Similarly a cut is taken on the face. The corresponding display position in Z-axis is noted down. The slide is taken back to a safe position and brought forward so that the turret face is touched on to component face. Corresponding display position in Z-axis is noted down.

The difference in the positions displayed corresponding to tool stationed at work zero and to that at the reference position is made use of to establish the tool length offsets.

NOTE:
When the slides are positioned at the machine reference point, turret zero itself is considered as the machine reference point.

Thus

\[
\begin{align*}
\text{X offset of tool} &= \text{X position at W/P} - \text{W/P radius} \\
&= 200 - 25 \\
&= 175
\end{align*}
\]

\[
\begin{align*}
\text{Z offset of tool} &= \text{Z position with the tool touching} - \text{Z position with table touching} \\
&= 100 - 92 \\
&= 8
\end{align*}
\]

The established data is entered into any one of the tool offset memory locations.
22.2 TOOL IDENTIFICATION CODE

Tool type is the tool identification code used for the purpose of application of tool nose radius compensation. There are nine types of tool identification codes from 1 to 9. The following diagram shows different location codes. Based on the position of tip centre with respect to leading tip the above codes are assigned.
23. **CUTTER RADIUS COMPENSATION:**
The tool offsets explained above are entered assuming that the cutting tip is sharp. But the actual cutting tip has a particular tip radius. The offsets taken in the above manner leads to contour errors whenever a contour machining is done.
On the OD the bottom most point, and on the face, the front most point of tool tip, touches the component.

Lines passing tangential to the “TIP CIRCLE” meet at point P. Tool tip touching at points 1 and 2 is as good as touching at point P. Keeping this point as reference CNC executes all tool movements. Point P is called as leading tip.

The example shown in the figure is having contour error.

Different programmed positions are attained by leading tip. It can be clearly seen from the drawing that the tool has already crossed point 1 and not yet reached point 2. Because of the leading tip there is profile error at the chamfer portion, i.e. chamfer is short.

CHAMFER IS LESS THAN THE PROGRAMMED VALUE, LEADING TO PROFILE ERROR
Since the cutting tip is of circular shape, depending on the contour profile, any point on the circumference may touch the component which leads to contour error. To avoid this sort of error tool radius compensation is used.

By programming radius compensation, control calculates and generates a new path called compensated path parallel to the programmed path, keeping the distance between the two paths equal to tip radius. Then the tool offsets get shifted to the tool tip centre and the tool tip centre follows the compensated path.

Compensated path being traced by the centre point, the counter error is eliminated. Programming of tool radius compensation is done by the relevant G code. There are two types of cutter radius compensation (C.R.C) like C.R.C to the left of the component G41 & C.R.C to the right of the component G42.

The above codes are decided looking in the direction of cutting, to which side of the component, the tool is existing i.e. if tool is to the left side, code is G41 and if the tool is to the right side, code is G42.

Looking in the direction of the feed if the cutting tool is to the right of the cutting surface code used is G42

Looking in the direction of the feed if cutting tool is to the left of the cutting surface code used is G41.

In the I figure, to effect C.R.C code programmed is G42 and in the II figure G41 is programmed.
PART PROGRAMMING MANUAL

DIRECTION OF CUTTING

Looking in the direction of the feed if the cutting tool is to the left of the cutting surface code used is G41

CODE PROGRAMMED IS G41

Looking in the direction of the feed if the cutting tool is to the right of the cutting surface code used is G42

CODE PROGRAMMED IS G42

Whenever C.R.C is selected, to initiate the compensation in the proper direction, CNC refers to the tool offset page to identify the tool location code and the tip radius.

G41 and G42 are modal. They are cancelled by G40 or after M30, reset or emergency stop.

23.1 SELECTION OF TOOL RADIUS COMPENSATION:
Tool radius compensation selection (G41 / G42) can only be carried out when G00 or G01 is active. Before selecting the tool nose radius compensation, the tool should have been called earlier (i.e. the tool programming with the relevant offset number is programmed earlier).
23.2 CANCELLATION OF TOOL RADIUS COMPENSATION:
Tool radius compensation cancellation (G40) can only be carried out in a block in which G00 or G01 is programmed.

Tool radius compensation cancellation is achieved by function G40. Tool offset memory number 00 corresponds to zero compensation. It can be used to cancel tool radius compensation.

G40, G41 and G42 can be programmed in a block separately with any other function. They are not active, unless a movement command is programmed at least in one axis.

When any change from G00 to G01, G02 or G03 is detected the CNC applies the same process as when the tool radius compensation is initiated.

24. G33, G34 THREAD CUTTING:
The following preparatory functions are available for cutting threads.
G33 thread cutting with constant lead / tapered threads / screw threads.
G34 thread cutting with linear lead increase / decrease

The spindle speed is read from the position coder on the spindle in real time and converted to the cutting feedrate for FPM, which is used to move the tool.

Spindle speed must remain constant from rough cutting through finish cutting, else incorrect thread lead will occur.

G33 (End point) (lead of the long axis):
IP    F    :

The range for thread lead is 0.0001 mm to 500.000mm
NOTE:
1. Feedrate override is not effective (fixed at 100X) during thread cutting.
2. If FEEDHOLD push button is pressed during thread cutting, the tool will stop after a block not specifying thread cutting is executed, similar to single block.
3. Spindle speed override is not effective in thread cutting mode.

24.1 THREADS WITH VARIABLE LEAD:
Threads with variable lead are cut with

\[
G34 \quad (\text{End point}) \quad (F) \quad (K)
\]

Lead in longitudinal axis direction at the start point
Increment / decrement of lead per spindle revolution.
Range for K is 0.0001 to 500.000 mm/rev.

24.2 MULTIPLE START THREADS:
Multiple start threads are cut by shifting the start point of the thread for each start, by the distance equal to pitch (lead divided by number of starts).

If the lead is 4mm and number of starts are 2, and if the start point for the first start of the thread is selected as Z4, then the start point for the second start of the thread is selected as either Z2 or Z6.
Chapter-4
M-Code List
## IV. M CODE LIST (New M-code list includes all optional features)

<table>
<thead>
<tr>
<th>CODE</th>
<th>MISCELLANEOUS FUNCTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>M00</td>
<td>Optional stop unconditional.</td>
</tr>
<tr>
<td>M01</td>
<td>Optional stop conditional</td>
</tr>
<tr>
<td>M02</td>
<td>End of program</td>
</tr>
<tr>
<td>M03</td>
<td>Clockwise direction of spindle rotation (viewing towards the chuck from machine front) of tailstock end.</td>
</tr>
<tr>
<td>M04</td>
<td>Anti-clockwise direction of spindle rotation (viewing towards the chuck from machine front) tailstock.</td>
</tr>
<tr>
<td>M05</td>
<td>Spindle stop</td>
</tr>
<tr>
<td>M07</td>
<td>Coolant ON</td>
</tr>
<tr>
<td>M09</td>
<td>Coolant OFF</td>
</tr>
<tr>
<td>M10</td>
<td>Chuck declamp</td>
</tr>
<tr>
<td>M11</td>
<td>Chuck clamp</td>
</tr>
<tr>
<td>M16</td>
<td>Chuck clamp on ID</td>
</tr>
<tr>
<td>M18</td>
<td>Chuck clamp on OD</td>
</tr>
<tr>
<td>M19</td>
<td>Oriented spindle stop</td>
</tr>
<tr>
<td>M20</td>
<td>Spindle orientation cancel</td>
</tr>
<tr>
<td>M21</td>
<td>30° rotation</td>
</tr>
<tr>
<td>M22</td>
<td>60° rotation</td>
</tr>
<tr>
<td>M23</td>
<td>90° rotation</td>
</tr>
<tr>
<td>M24</td>
<td>120° rotation</td>
</tr>
<tr>
<td>M25</td>
<td>150° rotation</td>
</tr>
<tr>
<td>M26</td>
<td>180° rotation</td>
</tr>
<tr>
<td>M30</td>
<td>End of program with return to beginning</td>
</tr>
<tr>
<td>M32</td>
<td>Tailstock quill forward</td>
</tr>
<tr>
<td>M33</td>
<td>Tailstock quill retract</td>
</tr>
<tr>
<td>M34</td>
<td>Parts catcher forward</td>
</tr>
<tr>
<td>M35</td>
<td>Parts catcher retract</td>
</tr>
<tr>
<td>M46</td>
<td>Door open</td>
</tr>
<tr>
<td>M47</td>
<td>Door close</td>
</tr>
<tr>
<td>M54</td>
<td>Tailstock quill override ON</td>
</tr>
<tr>
<td>M55</td>
<td>Chuck signal override ON</td>
</tr>
<tr>
<td>M56</td>
<td>T.S. quill interlock ON for cycles start</td>
</tr>
<tr>
<td>M57</td>
<td>T.S. quill interlock OFF for cycle start</td>
</tr>
<tr>
<td>M60</td>
<td>Door interlock OFF</td>
</tr>
<tr>
<td>M61</td>
<td>Door interlock ON</td>
</tr>
<tr>
<td>M82</td>
<td>T.S. body forward</td>
</tr>
<tr>
<td>M83</td>
<td>T.S. body retract</td>
</tr>
</tbody>
</table>
Chapter-5
Miscellaneous Functions
V. MISCELLANEOUS FUNCTIONS M:
Miscellaneous functions perform certain machine setting functions such as direction of spindle rotation, switching on the coolant motor etc., Each block may contain upto 3m functions. Some of the miscellaneous functions are defined by CNC system manufacturers and some are defined by machine tool manufacturers.

M00 PROGRAM STOP (UNCONDITIONAL):
When CNC reads the code M00 in a block, it stops executing the program. The START key is pressed to resume the program execution.

M01 PROGRAM STOP (CONDITIONAL):
It is same as M00, except that M01 is taken into account only if the optional stop input is activated. Optional stop input is activated by keeping OPT switch ON. (Selection with the soft key menu, OPT switch is made ON or OFF).

M02 END OF PROGRAM:
M01 indicates end of main program and performs a general reset function of the CNC.

M03 CLOCKWISE START OF SPINDLE:
By programming M03 the spindle is enabled to run in the clockwise direction.

M04 COUNTER CLOCKWISE START OF SPINDLE:
Similar to M03 except that the spindle is enabled to run in the counter clockwise direction.

M05 SPINDLE STOP:
By programming M05 spindle stop rotating

It is possible to change over from M03 to M04 and vice versa. In such a case before programming next M function it is necessary to program M05.

M07 COOLANT ON:
By programming M07, coolant motor i.e. low pressure coolant motor switches ON.

M09 COOLANT OFF:
By programming M09, coolant motor is switched OFF.

M10 CHUCK DECLAMP:
The function M10 means that the power chuck is actuated for declamping.
M11 CHUCK CLAMP:
The function M11 means that the power chuck is actuated for clamping.

M16 CHUCK CLAMP ON I.D:
By programming M16, the power chuck is actuated for internal clamping.

M18 CHUCK CLAMP ON O.D:
By programming M18, the power chuck is actuated for external clamping.

M19 ORIENTED SPINDLE STOP:
M19 facilitates to stop the spindle at a particular orientation with respect to zero marker on the spindle encoder. The angle is measured from the zero marker along the counter clockwise direction of rotation.

This feature is useful to drill offcentre holes, pitch holes at a particular p.c.d., radial slot milling on face, key way milling, oriented loading and unloading.

M20 SPINDLE ORIENTATION CANCEL

M21 30° ROTATION

M22 60° ROTATION

M23 90° ROTATION

M24 120° ROTATION

M25 150° ROTATION

M26 180° ROTATION

M30 MAIN PROGRAM END:
The function M30 is the same as M02. After resetting the internal registers the CNC control returns the cursor to the top (1st block) of the currently active part program.

M32 TAILSTOCK QUILL FORWARD:
By programming M32, tailstock quill could be brought forward.

M33 TAILSTOCK QUILL RETRACT:
By programming M33, tailstock quill is retracted.
M34 PARTS CATCHER FORWARD
M35 PARTS CATCHER RETRACT
M46 DOOR OPEN
M47 DOOR CLOSE
M54 TAILSTOCK QUILL OVERRIDE ON
M55 CHUCK SIGNAL OVERRIDE ON
M56 T.S. QUILL INTERLOCK ON FOR CYCLE START
M57 T.S. QUILL INTERLOCK OFF FOR CYCLE START
M60 DOOR INTERLOCK OFF
M61 DOOR INTERLOCK ON
M82 T.S. BODY FORWARD
M83 T.S. BODY RETRACT
CANNED CYCLES - G71 ~ G76 (MULTIPLE REPETITIVE CYCLE):
G71 STOCK REMOVAL IN TURNING.
G71 U_ R_ ;
G71 P_ Q_ U_ W_ F_ ;
G71 UΔd Re;
G71 Pns Qnf UΔu WΔw Ff;

Δd = Depth of cut in radius
e = Tool escape / tool retraction distance
ns = Sequence number of the first block of the program which specifies the finish figure.
nf = Sequence number of the last block of the program which specifies the finish figure.
Δu = Finish allowance on “X” axis / diameter.
Δw = Finish allowance on “Z” axis / face.
f = Feed

Example:

--- Diagram ---

Raw material φ75x105

R3

R2 45°

R2

R3

φ70

φ60

φ40

φ25

X

75

60

25

1x45°
O00 71 ;
N1 T00 00 ;
G00 X0. Z-100.;
T0606 M07 ;
G92 S2500 ;
G96 S180 M03 X75.Z0. ;
G01 X-1 F0.25 ;
G00 X75.Z2. ;
G71 U2.5 R1. ;
G71 P10 Q20 U1. W0.5 F0.3 ;
N10 G0 X21. ;
G01 G42 X23. Z0. F0.2 ;
A135. X25. ;
Z-25. ;
X40. R3. ;
A180. R2. ;
X60. Z-60. A135. R2. ;
Z-75. R3. ;
X72. 
N20 G00 GG40 X75. ;
G70 P10 Q20 ; FINISHING CYCLE
N2 S0 T00 00 ;
G00 X0.Z-100. ;
T0808 M07 ;
G92 S2500 ;
G96 S200 M03 X75. Z2. ;
G70 P10 Q20 ;
S0 T00 00 M09 ;
G0 X0.Z-100. ;
M05 ;
M30 ;
G72 STOCK REMOVAL IN FACING:

G72 WΔd Re
G72 Pns Qnf UΔu WΔu Ff

Δd = Depth of cut in Z axis.

e = Tool escape / retract distance.

ns = Sequence number of the first block of the program which specifies the finish figure.

nf = Sequence number of the last block of the program which specifies the finish figure.

Δu = Finish allowance on “X” axis / diameter.

Δw = Finish allowance on “Z” axis / face.

f = Feed
EXAMPLE:
O0072;
T0000;
G0 X0. Z-100. M07;
T0101
G92 S2500;
G96 S180 M03 X105. Z0. ;
G01 X-1. F0.25;
G00 X105. Z2. ;
G72 W1.5 R1. ;
G72 P10 Q20 U0.5 W0.25 F0.3 ;
N10 G0 Z-15. ;
G01 X60. Z-10. ;
X30. ;
X25. Z0. ;
N20 X0. ;
G97 S0 T0000 M09 G0 X0. Z-100. ;
M30 ;

G73 – PATTERN REPEATING CYCLE:
G73 UΔI WΔk Rd;
G73 Pns Qnf UΔu WΔw Ff. ;
G73 U_ W_ R_ ;
G73 P_ Q_ U_ W_ F_ ;

ΔI = Machining stock on “X” axis / machining allowance on “X” axis.
ΔK = Machining stock on “Z” axis.
d = Number of rough cuts.
ns = Sequence number of the first block of the program which specifies the finish figure.
nf = Sequence number of the last block of the program which specifies the finish figure.
Δu = Finish allowance on “X” axis / diameter.
ΔW = Finish allowance on “Z” axis / face.
f = Feed
G74 – PECK DRILLING CYCLE:
G74 R(e) ;
G74 Z(W) Q(Δk) F(f) ;
Re = Return amount / relief depth
ZW = Depth of drill hole
QΔK = Incremental depth / depth of peck
Ff = Feed

EXAMPLE:

O0074 ;
T000 ;
G00 X0 Z-100. ;
S900 M04 ;
T0505 ;
G00 X0 Z3. M07 ;
G74 R1000 ;
G74 Z-60. Q20000 F0.2 ;
G00 Z5. ;
S0 T0000 M09 ;
X0 Z-100. M05 ;
M30 ;
G75- GROOVING CYCLE ON “X” AXIS:
G75 R(e)
G75 X(u) Z(w) P(∆i) Q(∆k) R(∆d) F(f)

- e = Tool return amount / relief depth
- u = Groove diameter
- w = Width of groove / end point of groove
- ∆i = Incremental depth of cut on “X” axis
- ∆k = Incremental depth of cut on “Z” axis or tool advancement in “Z” axis.
- ∆d = Relief amount of the tool at the cutting bottom
- f = Feed

Example:

```
O 0075 ;
T0000 ;
G00 X0. Z-100. M07 ;
T0101 S300 M03 ;
G92 S2000 ;
G96 S100 X82 Z-10. ;
G75 R1000 ;
G75 X50. Z-21. P5000 Q4000 F0.15 ;
G00 X82. ;
S0 T00 00 X0. Z-100. M09 ;
M05 ;
M30 ;
```

Tool width = 4mm
G76 – MULTIPLE REPETITIVE PASS THREADING – CYCLE :
G76 P m r a Q Δd min. R d ;
G76 X u Z w R i P k QΔd F l ;
G76 P_ _ _ Q_ R_ ;
G76 X_ Z_ R_ P_ Q_ F_ ;

m = Number of finish passes (1-99)
r = Chamfering amount (01-99) 01= 0.1, 99 = 9.9
(a=0.1x1 to 9.9x1)
a = Angle of tool tip / angle of thread. (0°, 60°, 55°, 30°, 29°)
d = Finish allowance
u = Minor diameter of the thread in case of external thread – Major diameter in case of internal.
w = Length of thread.
i = Difference of thread radius (for taper threads only)
k = Height of thread / depth of thread
l = Lead or pitch of thread

(Δd min) = minimum depth of cut, subsequent passes depth (Δd√n - Δd√n-1). Number of cutting passes can be controlled by varying this value.
Δd = Depth of cut in 1st pass / 1st cut.

Example :
M20x2.5
Pitch = 2.5

EXTERNAL THREADING – M20 :
O000076 ;
T0000 ;
G00 X0. Z-100. ;
T0808 M07 ;
G97 S1000 M04 ;
G00 X22. Z5. ;
G76 P020060 Q200 R100 ;
G76 X16.8 Z-40. P1600 Q500 F2.5 ;
S0 T0000 ;
G00 X0. Z-100. M09 ;
M05 ;
M30 ;

INTERNAL THREADING – M20 :
O0079 ;
T0000 G00 X0 Z-100. ;
T0707 M07 ;
G97 S1000 M04 ;
G00 X15. Z5. ;
G76 P020060 Q200 R100 ;
G76 X20. Z-40. P1600 Q500 F2.5 ;
S0 T0000 X0. Z-100. M09 ;
M05 ;
M30 ;
MULTIPLE START THREADING:
Four start thread cutting on φ30mm (RH) lead 8mm
Pitch = \frac{\text{Lead}}{\text{No. start}} = \frac{8}{4} = 2\text{mm}

O0763 ;
T0000 ;
G0 X0. Z-100. ;
T0505 M07 ;
G97 S1000 M04 ;
G0 X32. Z10. ;
1 G76 P020060 Q150 R50 ;
   G76 X27.44 Z-60. P11280 Q400 F8.0 ;
   G0 X32.Z8. ;
2 G76 P020060 Q150 R50 ;
   G76 X27.44 Z-60. P1280 Q400 F8.0 ;
   G0 X32.Z6. ;
3 G76 P02.0060 Q150 R50 ;
   G76 X27.44 Z-60. P1280 Q400 F8.0 ;
   G0 X32. Z4. ;
4 G76 P020060 Q150 R50 ;
   G76 X27.44 Z-60. P1280 Q400 F8.0 ;
   S0T0000 M09 ;
   G0 X0.2-100. M05 ;
   M30 ;
TAPPING OPERATION:

O5555;
T0000;
G0 X0.Z-100;
T0303 M07;
G97 S200 M04;
G0 X0. Z5.;
G33 Z-20. F1.75 ; / G01 Z-20. F1.75 ;
S0;
S200 M03;
G33 Z5. F1.75 ; / G01 Z5. F1.75 ;
S0 T0000 M09;
G0 X0.2-100.;
M05;
M30;
DIRECT DIMENSION PROGRAMMING:

A – ANGLE

1. Always specified positive (i.e. in the counter clockwise keeping Z-axis as base).
2. Specifies the direction of cutting.

C-CHAMFER

The extent of corner break for each side

R – CORNER RADIUS
1. **ONE OF THE 2nd POINT CO-ORDINATES IS MISSING:**

   
   \[ X^+ \]
   
   \[ 2X_2(\text{or } Z_2) \]
   
   \[ 1X_1Z_1 \]
   
   \[ Z^+ \]

   N10 \(X_1Z_1\);
   
   N15 \(X_2(\text{or } Z_2)\) \(A\).

2. **INSERTION OF CHAMFER AT THE 2nd POINT:**

   
   \[ X^+ \]
   
   \[ 3X_3Z_3 \]
   
   \[ X_2(\text{or } Z_2) \]
   
   \[ 2 \]
   
   \[ 1X_1Z_1 \]
   
   \[ Z^+ \]

   N10 \(X_1Z_1\);
   
   N15 \(X_2(\text{or } Z_2)\) \(A \) \(C\);
   
   N20 \(X_3Z_3\).

3. **INSERTION OF RADIUS AT THE 2nd POINT:**

   
   \[ X^+ \]
   
   \[ 3X_3Z_3 \]
   
   \[ X_2(\text{or } Z_2) \]
   
   \[ 2 \]
   
   \[ 1X_1Z_1 \]
   
   \[ Z^+ \]

   N10 \(X_1Z_1\);
   
   N15 \(X_2(\text{or } Z_2)\) \(A \) \(R\);
   
   N20 \(X_3Z_3\).
4. **BOTH OF THE 2nd POIT CO-ORDINATES ARE MISSING:**

X+ 3X3Z3

\[ N10 \ X1 \ Z1 ; \]
\[ N15 \ A1 ; \]
\[ N20 \ X3 \ Z3 \ A2 ; \]

5. **INSERTION OF A CHAMFER AT THE 2nd POINT:**

X+ 3X3Z3

\[ N10 \ X1 \ Z1 ; \]
\[ N15 \ A1 ; \]
\[ N20 \ X3 \ Z3 \ A2 ; \]

6. **INSERTION OF A RADIUS AT THE 2nd POINT:**

X+ 3X3Z3

\[ N10 \ X1 \ Z1 ; \]
\[ N15 \ A1 \ R ; \]
\[ N20 \ X3 \ Z3 \ A2 ; \]
7. INSERTION OF CHAMFER AT 2\textsuperscript{nd} AND 3\textsuperscript{rd} POINT:

\begin{align*}
\text{N10} & \ X_1Z_1; \\
\text{N15} & \ Z_1C_1; \\
\text{N20} & \ X_3Z_3A_2C_2; \\
\text{N25} & \ X_4Z_4; \\
\end{align*}

8. INSERTION OF CHAMFER AT 2\textsuperscript{nd} AND RADIUS AT 3\textsuperscript{rd} POINT:

\begin{align*}
\text{N10} & \ X_1Z_1; \\
\text{N15} & \ A_1C_1; \\
\text{N20} & \ X_3Z_3A_2R_2; \\
\text{N25} & \ X_4Z_4; \\
\end{align*}

9. INSERTION OF RADIUS AT 2\textsuperscript{nd} AND 3\textsuperscript{rd} POINT:

\begin{align*}
\text{N10} & \ X_1Z_1; \\
\text{N15} & \ A_1R_1; \\
\text{N20} & \ X_3Z_3A_2R_2; \\
\text{N25} & \ X_4Z_4; \\
\end{align*}
10. **INSERTION OF RADIUS AT 2\textsuperscript{nd} POINT AND CHAMFER AT 3\textsuperscript{rd} POINT**:

\begin{align*}
N10 & \ X_1Z_1; \\
N15 & \ A_1 R_1; \\
N20 & \ X_3 Z_3 A_2 C_2; \\
N25 & \ X_4 Z_4; \\
\end{align*}

\begin{align*}
\text{C}_2 & \ X_3 Z_3 \\
3 & \ 2 \\
R_1 & \ A_1 \\
1X_1Z_1 & \ Z^+ \\
\end{align*}